CNC Milling on Wells/Index Vertical Mill
with Heidenhain Controller

Step One: Transfer CNC program from PC to Heidenhain controller

1. Heidenhain in Edit Mode  Press,  Use arrows to select READ-IN SELECTED PROGRAM, Press 
2. Setup the PC to send the program. D:\PC-HEID\ HEID.BAS
3. <F2> and type in filename, including drive letter (3.5in. Floppy is Drive B)
4. <F1> to select Cut Only, then press <CR>
5. On Heidenhain, enter a program number and press 
6. On PC, press <CR>. Transfer should begin. When transfer is complete, the program should be displayed on the Heidenhain CRT. If the message WRONG BLOCK appears, then press move the cursor to the end of the program and delete the out-of-sequence blocks. Press <CR> on PC to end the transfer program and type SYSTEM <CR> to get back to the regular screen.

Step Two: Edit CNC program on Heidenhain and run test.

1. Make the new program active by pressing  , enter program number and press 
2. Switch to  , Press  , enter 1 the 
3. Switch to Test Mode,  Enter 9999 for the Block Number, press  . The test will stop at tool changes and you must press  to continue on through the program. Watch the I and J coordinates at the bottom of the CRT. They should change as the program runs. Note the block number of any errors and the error message. Make necessary corrections and repeat process until the test runs without any errors. This may involve regenerating the toolpaths on I-DEAS.

Step Three: Enter tooling parameters into program

1. Switch to Edit Mode, press  and adjust the tool length parameters for each tool to be used in the program. Tool definitions are given in G99 blocks. Tool #1 is the zero length tool, so you only have to specify a radius for this tool if you wish to run the graphics mode. For tools #2-9 (as needed) enter the length offsets and cutter radii. The length offset is negative for tools that are shorter than tool # and positive for longer tools.

Example: Tool #2 is a ½ in. end mill that is 0.145 in. shorter than tool #1.,
Original block → **N60 G99 T2 L+0. R+0.**

Edit to look like → **N60 G99 T2 L-0.145 R+0.125**

Step Four: Graphics Preview

1. Make sure that the starting stock size has been defined near the beginning of the program with G30 and G31 codes. This should be around blocks 50-55.
2. Switch to  ,  , 1, 
3. Under the GRAPHICS buttons, press  and 
4. Press  as needed when blocks stop incrementing to continue generating the part. *Don't press start at the end of the job or it will start over again!*
5. Review the graphic image of the part to make sure it agrees with the drawings of the part.

Step Five: Machine the part.

1. Position the fixtures and starting stock as indicated on the supplied drawings.
2. Set the datum based on the zero datum indicated on the drawings. Note that the position for tool changes in X=-5, Y=0, Z=1. Make sure that the table cn reach this position when setting up the fixtures.
3. It is highly recommended that you cut air or a foam blank with the program before machining any metal. Also, when performing a dry run, make sure to keep a hand near rthe EMERGENCY STOP button in case the machine tries to crash into a fixture or the table.
4. Tool changes are indicated by the yellow TOOL CHG light. Press this button and then SPINDLE STOP to change the tool. Tool changes are made by switching to Manual mode and jogging the quill to the fully up position. After the tool is changed, switch back to Full Seq. Run,  and press SPINDLE CW and CYC START.

R.E. Link
Mechanical Engineering
March 1997